DRG
Dynaflow Research Group

BOSfluids

Tutorial

Getting Started

The “Getting Started” tutorial guides the user through the process of building a model, defining boundary conditions and the basics of post-processing. This tutorial is designed for people who are not familiar with the software and shall demonstrate the basic features.

1. CONSTRUCTING THE MODEL

1.1. Introduction

The following system will be used to demonstrate the use and some of the basic features of the transient fluid flow simulation package BOSfluids®. The example consists of a simple water evacuation system that is 80 meters long and has a gate valve at the end that closes in 0.5 seconds. The steel piping has a roughness = 0.05 mm. The analysis should predict the pressure rise in the system upon valve closure.

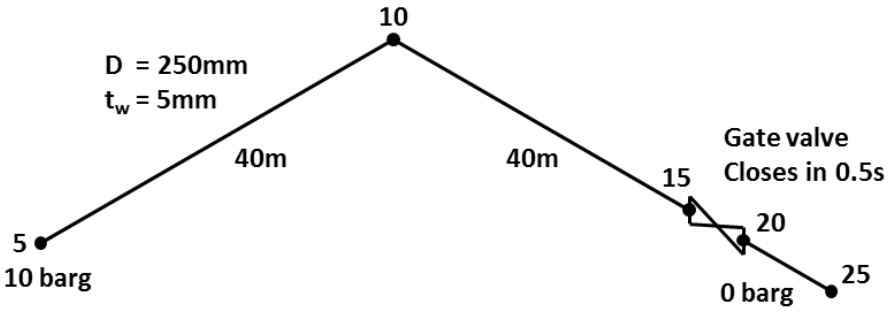

Figure 1 | Example of a water evacuation system

1.2. Starting the program and creating a new model

Start the program BOSfluids. Using the *New Job* button, a new model can be created. Name the new model *GetStarted* and select *Metric* in the unit selection box. If a model has already been created, use the *Open Job* button and select the file (with extension *.bfz*).

Figure 2 | BOSfluids Start-up screen

When the program has opened, it shows the first tab; *Piping*. Here the piping layout can be entered one element at a time. Another possibility is to import an existing piping layout

Copyright © Dynaflow Research Group.

from a pipe stress software package such as CAESAR II, or from a previous version of BOSfluids. Import features are discussed in other tutorials.

The construction procedure of the model, depicted in Figure 1, will be described in detail in this tutorial. During entry of data, the user can get help by pressing *Shift + F1*, or selecting *Help → What's This?* from the toolbar and select the input field of interest.

To enter the first element, press the "create new element" icon, **+**. Set the element type to *Pipe* in the input screen. This changes the *dynamic input screen*, to show the required input parameters applicable to this element type. Fields that are still required to be filled out are displayed in red. Change the first node number to 5 and the second node number to 10. Now, create a 40m long pipe in the negative Z-direction with diameter 250mm, wall thickness 5mm and pipe roughness 0.05mm. The input data for pipe material, pipe restraint and fluid temperature can be left at their default values, see Figure 3. Click *Apply Changes*, or press enter to confirm the input.

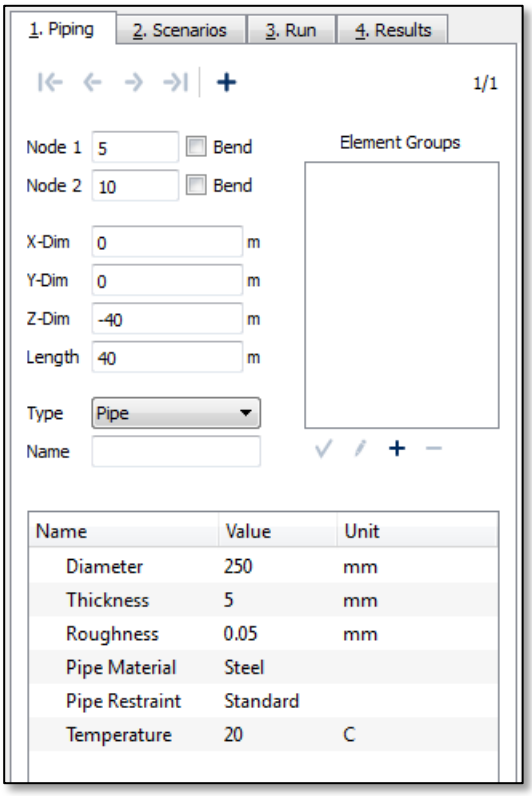

Figure 3 | Input screen for the first element

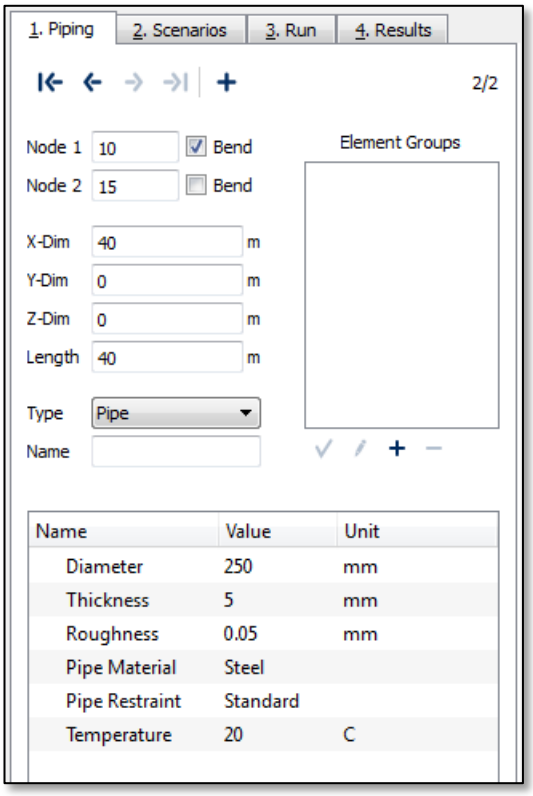

Figure 4 | Input screen for the second element

Now create the second element by clicking the **+** button. The element type is now set automatically to the previous *Pipe* element. Change the second node number to 15 and create a 40m long pipe element in the X-direction. The element parameters are the same as defined in the first element and can be left unchanged.

Copyright © Dynaflow Research Group.

At node number 10 there is a bend in the pipe system. This can be easily incorporated by ticking the *Bend* box next to Node 1, see Figure 4. When navigating through the elements using the arrows at the top of the tab, it can be seen that node number 10 in the first element is now also assigned to be a bend.

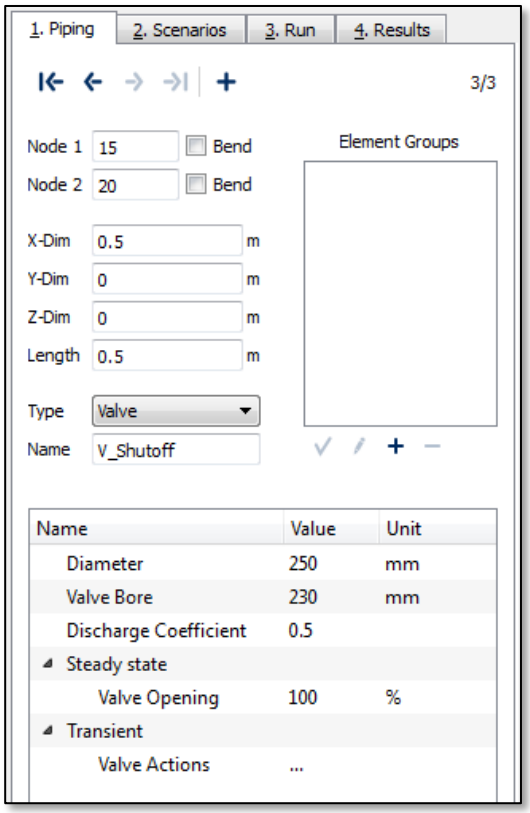

Figure 5 | Input screen for the third element

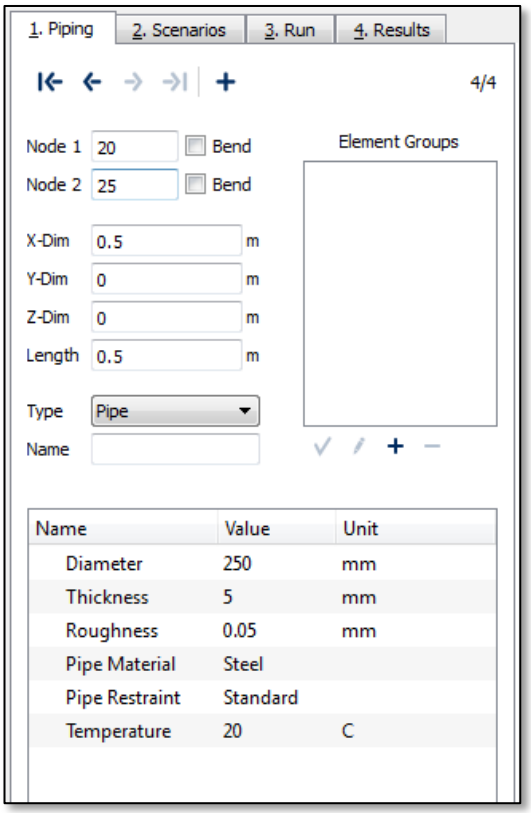

Figure 6 | Input screen for the fourth element

The third element is of type *Valve* and is 0.5m long in the X-direction, see Figure 5. Select *Valve* from the Type drop down menu, set the *Valve Bore* to 230mm and the *Discharge Coefficient* to 0.5. Check that the option *Steady State* → *Valve Opening* is set to 100%, to have the valve fully open during steady state analysis.

Each element can also be given a *Name*. This is especially useful for special components like pumps, valves, etc. BOSfluids generates default names for all elements other than piping element. Name the valve element V_Shutoff.

Transient actions such as a valve closure are found under *Transient* → *Valve Actions*. By double clicking the … button next to *Valve Actions*, the valve opening as function of time can be entered. For this example, the valve closes linearly from 100% open at time 0 seconds to fully close (0% open) at time 0.5 seconds, see Figure 7.

By default the flow coefficient, which determines the resistance the flow experiences when passing through the valve (resulting in a pressure loss over the valve), varies linearly with the valve opening. For this case the flow coefficient at 100% opening is specified by the

Copyright © Dynaflow Research Group.

Discharge Coefficient and has a value of 0.5. Using the *Configure Valve* window the flow coefficient can also be specified by giving a Cv or Kv value. Also a custom valve curve can be specified, but in this example we keep these settings as default.

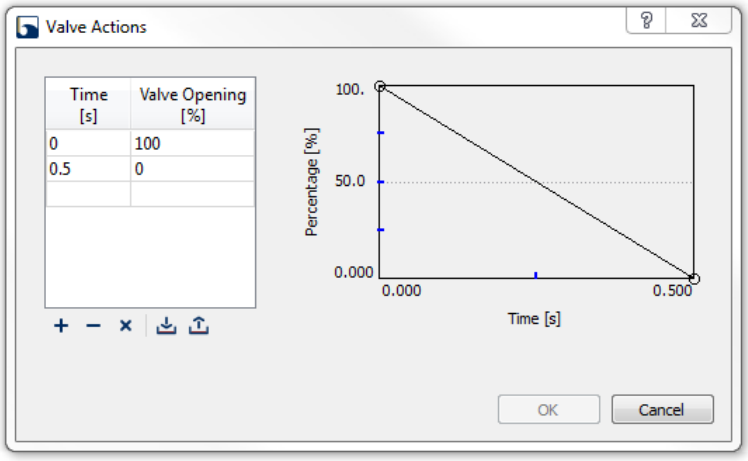

Figure 7 | Valve Action screen

Finally add the last element again of type *Pipe*. Change the second node number to 25 and create a 0.5m long pipe element continuing in the X-direction. The element parameters are the same as defined in the other pipe elements, see Figure 6.

The basic construction of the piping model is now finished. The *Edit* menu or right clicking the piping model in the 3-D viewer provides all kinds of options to insert delete or split elements when required.

1.2.1. Units

Different sets of units can be selected. At the top of the screen, go to *Tools -> Units* and select the preferred type of units. English units as well as SI units are available. Switching units can be done at any time; at the start of a model but also during the modeling or even at the analysis phase.

1.2.2. Visualization

The model is depicted in the *3D Visualization Screen*. The view can be altered by using the buttons *Pan, Rotate, Zoom* and *Center* at the top of the visualization screen. Because the length of pipe elements can be very large compared to the diameter, details like valves might be difficult to view. A useful tool is the *Diameter Multiplier*, which can be found by clicking the visualization settings button at the top-right of the visualization screen . Setting this to a larger value makes components and details in the routing, like bypasses or loops, more visible.

Depending on your screen resolution, set the *Diameter Multiplier* to a larger value, for example 5. Pressing the show labels button 10 will show the node numbers and element

Copyright © Dynaflow Research Group.

names (where used). The model should now look like Figure 8. The user can also select to view the model as a *Wireframe* or select *Search* to look for a specific node or element. The measurement tool can be used to show the absolute distances between two nodes and also shows the shortest possible path between them.

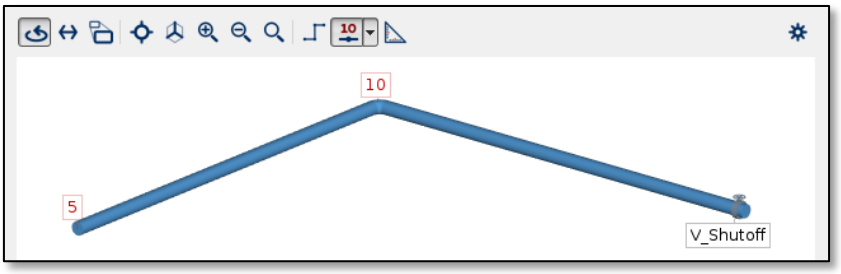

Figure 8 | The model shown in the *visualization screen* including name/node tags

2. DEFINING THE PARAMETERS

Switch to the second tab: *Scenarios*. In this tab the parameters for the simulation are specified. The parameters are specified for at least one scenario: *Main*. This is the base scenario and all simulation parameters need to be specified for this scenario.

However, the user can also generate additional scenarios by applying small changes to the main scenario, without the need to create multiple variations on the same model in separate files. In this way, small variations can be easily analyzed without losing track of the main scenario.

Note that when multiple scenarios are created the *Main* scenario will serve as a basis for all other scenarios. Changes made to the *Main* model, will propagate through to all the scenarios, overwriting the corresponding values in each scenario. This prevents the need to alter all the scenarios if a major change is required. The *Main* scenario can therefore not be deleted.

2.1. Element Parameters

In the first sub-tab *Elements*, the element parameters are defined. For each element all parameters including the element type can be altered. The element parameters for the *Main* scenario can also be adjusted in the *Piping* tab.

Elements can be made part of an element group. Element groups are created automatically per element type; all piping elements are added to element group *Pipe*, all valves are added to element group *Valve*, etc. Element groups can also be created manually by clicking the **+** button at the bottom of the *Element Groups* list. An element can be made part of more than one group.

Copyright © Dynaflow Research Group.

For larger systems, a smart use of element groups makes it very easy to change the input data at a later stage of a project. The element groups allow for quick input data manipulation by selecting an *Element Group* and then selecting all or some different elements in the *Elements* list. In the *dynamic input screen* the input parameters appear that are applicable to all elements. For example, when pipe elements are selected, all input fields that contain the pipe parameters appear. Changing a parameter in the *dynamic input screen* will apply the new value to all selected elements. When different types of elements are selected, only those parameters appear in the *dynamic input screen* that are common to all selected element types, except when the option *Show All Fields* has been enabled.

2.2. Node properties and boundary conditions

In the sub-tab *Nodes & BCs*, various node properties can be entered.

By selecting a node in the *Nodes* list on the right, it is possible to change the *Node Type* and the node properties. Selecting Node 10 in this example will show that the *Node Type* is *Bend*. The *Bend Radius* parameter can be changed here. The default value is 1.5 times the pipe diameter. For the current example, this value is correct.

Similar to the elements and element groups, all nodes are automatically placed in a *Node Group* based on the node type. When selecting a node group in the *Node Groups* list all nodes belonging to that group are shown in the *Nodes* list. Again it is also possible to manually create a group.

Make a new group by selecting nodes 5, 10 and 15 and clicking the *Add To Group* button, see Figure 9. An *Add To Group* window will appear, with all custom groups shown. The user can create a new group by selecting the **+**. Name the group PipeNodes and optionally provide a brief description.

2.2.1. Defining boundary conditions

The system is subjected to the following boundary conditions:

- fixed reservoir pressure of 10 barg at node 5
- open-end atmospheric pressure of 0 barg at node 25

Note: All pressures entered are *gauge* pressures, i.e. overpressures with respect to the ambient pressure. The entered pressure should not include any pressure due to fluid elevation.

Enter the boundary conditions by clicking node group *All,* followed by clicking node 5. Now, choose *Node Type = Fixed Pressure.* At the *dynamic input screen,* enter 10 barg as the prescribed pressure, see Figure 10. Do the same for node 25 and enter 0 barg.

Copyright © Dynaflow Research Group.

BOSfluids has now created a Node Group *Fixed Pressure*. All nodes with fixed pressure boundary conditions will be automatically added to this group. Switching to this group will show both node 5 and node 25.

To remove a boundary condition from a node, select the node and change the *Node Type* to *Simple*. Simple is the default node type, indicating that no boundary condition is defined at that node.

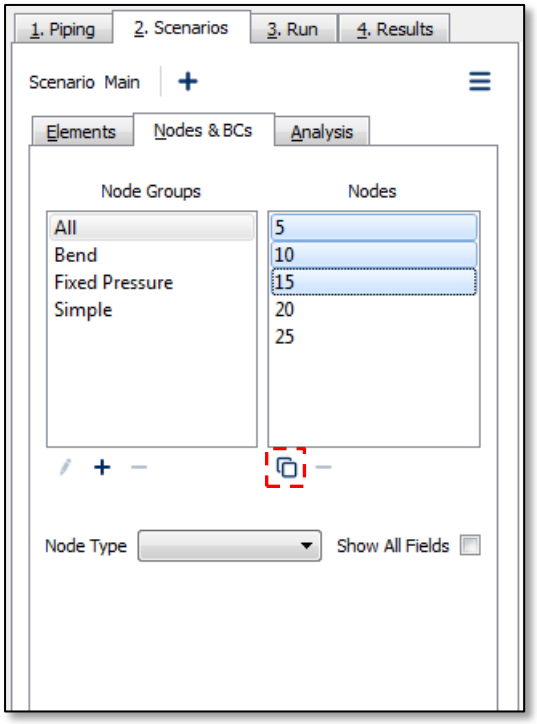

Figure 9 | Create a custom *Node Group* by selecting nodes and clicking the *Create Group* button

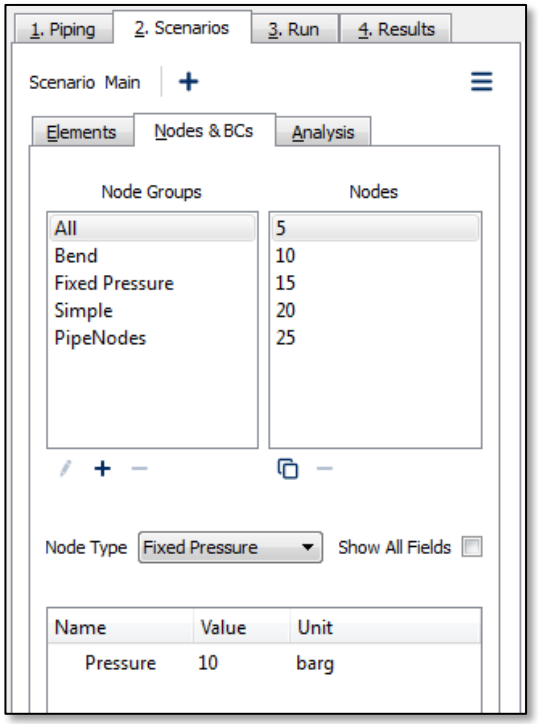

Figure 10 | Create boundary conditions at Node 5 and 20

Boundary conditions can be provided either as pressure or as flow rate, but to create a solvable solution the pressure must be specified at least one location.

2.3. Setting up an analysis

At the sub-tab *Analysis*, the analysis type can be selected. This can be S*teady State*, *Transient*, *Transient VCM*, or *Transient CAP* (the last two include a cavitation model). Based on the selected analysis type, the *dynamic input screen* shows parameters that can be set for this analysis. A description of each parameter can be obtained by using the *Help* function or by consulting the *User Manual*.

For the current example, select *Analysis Type* → *Transient* and keep *Fluid Type* → *Water*. All other parameters can be kept at their default values, see Figure 11.

Copyright © Dynaflow Research Group.

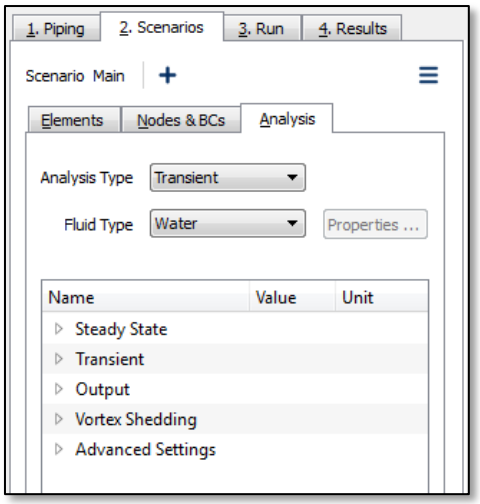

Figure 11 | Input screen for Analysis

2.3.1. Initial flow conditions

Before a transient analysis is performed, the program needs to know the initial flow conditions. Typically a transient analysis is performed using the results of a steady state analysis as initial flow conditions, but there is also an option to start the transient analysis from a no-flow state. For the first case, the steady state analysis is automatically performed before the transient analysis is performed.

2.4. Scenarios

As explained at the beginning of this section, different scenarios can be created by applying small changes to the main model.

For the current example, we include a scenario which uses a higher pressure boundary condition. Click on the menu icon ☰, or select *Tools* → *Scenarios* from the window toolbar. A *Scenarios* window will appear, currently showing only *Main*. By clicking the *add a new scenario* icon, **+**, the user can add a new scenario. Name this scenario *Higher Pressure* and press *OK*. Now multiple scenarios can be selected. Adding a new scenario by using **+**, will make a copy of the main scenario. In the *Scenarios* window also other scenarios can be copied or the differences between two scenarios can be shown.

Select the *Higher Pressure* Scenario. For the new scenario element, node and analysis properties can be changed. However, it is not possible to change the geometry of the model. Go to the sub-tab *Nodes & BC* and change the boundary condition at Node 5 to be 15 barg, see Figure 12.

The model is now ready to be run.

Copyright © Dynaflow Research Group.

3. RUNNING THE SIMULATION

Go to the tab *Run*. The scenarios for which a run must be performed can be selected from the *Scenario List*. Tick both scenarios and click the *run* button, indicated in Figure 13.

Any relevant messages, like errors or warnings, found in the model during a run, appear at the *Messages* section and the runs that have been completed appear in the list *Completed*.

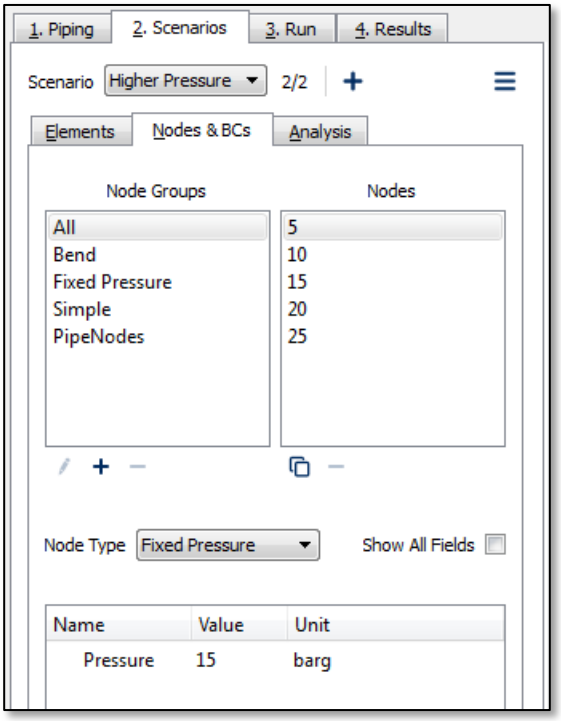

Figure 12 | Create a custom second *Scenario*, where the pressure at Node 5 is higher

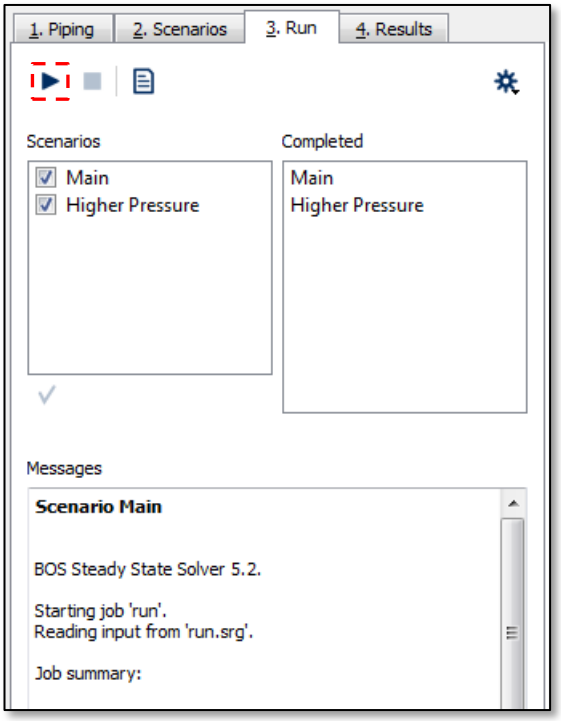

Figure 13 | Start the simulation and run both scenarios

4. REVIEWING THE RESULTS

Select the tab *Results.* Here the output of the simulation can be reviewed. The results can be reviewed in the 3-D viewer, where the complete system can be shown for each time step. The results can also be shown as 2-D plots, in which the fluid properties per node or element are shown versus time.

When performing a steady state analysis there is no variation in time. This means that the 3-D output is limited to time = 0 and no 2-D output is available.

4.1. 3-D Output

Under the Heading 3-D Output, the user can select the scenario of interest and choose from the various data sets; *Flow Rate, Force, Max Pressure, Pressure* and *Velocity*. The selected data set is directly plotted in the 3-D viewer. To review the results through time, the slider can be

Copyright © Dynaflow Research Group.

used, a time location can be entered, or the results can be played automatically by clicking the *Animate* button ▶. The speed of the animation can be changed using the *Set the animation speed* button. When the time is set at 0 seconds the values shown correspond with the steady state solution.

The minimum and maximum values for the whole simulation can be shown in the 3-D viewer by selecting this option under the *Configure Output Settings* button ⚙. The values themselves can be shown using the *Data Set* information button.

4.1.1. Profile plot

A profile plot can be generated by clicking the button. A first and last node needs to be entered and the selected variable in the *Data Set* is plotted along the length of the piping. Note that when entering a first and last node, the user needs to select a unique flow path. For flow paths with loops, the program will return a message that no logical graph output is possible.

The *Profile plot* and the *3-D graphical results* can be viewed simultaneously. When using the slider or the *Animate* button, both the *3-D Visualization* and the *Profile plot* vary through time in a synchronous way.

Alternatively also a *Min-Max plot* can be generated that shows all nodes arranged based on their minimum and maximum values

4.2. 2-D Output

Under the heading *2-D Output*, users can select a data set of interest and plot this for selected points of interest within the system. The available data sets depend on the type of element/node.

- **Nodes** – For individual nodes, you can plot *Flow Rate, Force, Max Pressure, Pressure* and *Velocity.*
- **Node groups** – For groups of nodes, you can plot the same variables as for individual nodes. The names of the group correspond with the groups specified in the *BCs and Nodes* sub-tab.
- **Element** – For all element types different than the pipe element, you can plot the variables of interest. For example, for a valve, you can plot *Flow Rate, Opening* and *Pressure Drop.*
- **Element groups** – For groups of elements, you can plot the same variables as for individual elements. The group name corresponds with the groups specified in the piping sub-tab.

Copyright © Dynaflow Research Group.

The selected variable is plotted against time (or frequency) on the horizontal axis. When plotting groups of nodes or elements, the results for each node or element in the group are plotted on a single graph. The plot type can be changed from *Time Domain* to *Frequency Domain* to show a frequency breakdown of the selected variable.

4.3. Interpretation of the results

In the *2-D Output* select *Node Group* → *PipeNodes*, tick the *Main* scenario and *Pressure* data set, and click plot. The transient variation in the pressure at nodes 5, 10 and 15 is shown in Figure 14.

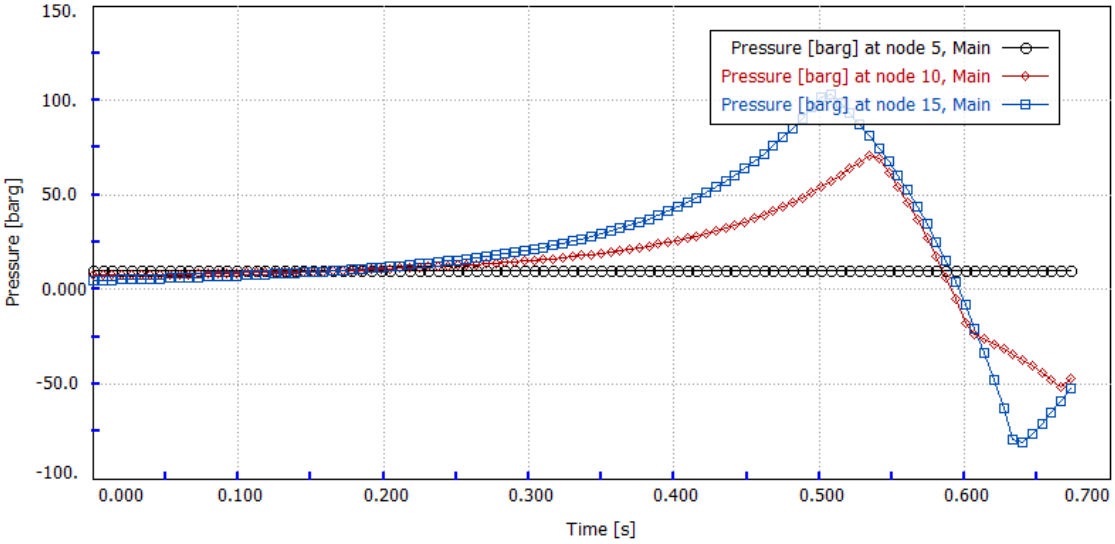

Figure 14 | Results for the *Pressure* variation through time for each pipe node

In Figure 14, two results stand out. The first result that should be noticed when studying Figure 14 is the negative pressures that are reached after 0.6 seconds. The unphysical negative pressures (full vacuum is equal to -1 barg) occur when an expansion wave progresses through the fluid. In reality vapour formation and column separation will occur when the line pressure drops below the fluid vapour pressure.

The second result to notice in Figure 14 is that the results are only shown until 0.7 seconds, while the interesting period is longer.

To solve these issues we need to go back to the tab *Scenarios*.

4.3.1. Redefining the analysis parameters

BOSfluids can include the effect of vapour formation by selecting the *Concentrated Air Pocket* (CAP) cavitation analysis model.

Return to the *Main* scenario and select the sub-tab *Analysis*. Set the *Analysis Type* to *Transient CAP* from the drop down menu. This analysis type will include the effect of cavitation and

Copyright © Dynaflow Research Group.

vapor formation in the simulation. The change in the *Main* scenario will also update the *Higher Pressure* scenario.

Extra *CAP options* are now available. The *CAP options* require the parameter for *Void Fraction* to be filled out. This parameter represents the volumetric fraction of gas for the Reference pressure conditions (leave this at Atmospheric Pressure for this example). Use the value $5.5 \cdot 10^{-6}$ for this parameter, see Figure 15. The program uses the *Void Fraction* to model the growth and collapse of gas bubbles in the liquid. The CAP model prevents the pressure to drop below vapour pressure and includes the possibility of pressure spikes due to bubble collapse.

The end time of a simulation is by default determined automatically based on the system properties and transient actions. In some cases the interesting events may take place after the transient actions are finished. For these cases the end time can also be set manually.

Under *Transient* options change the *Simulation Time* to 5 seconds. This allows the pressure pulsations originating from the valve closure to reach the end of the system and reflect back into the direction of the valve.

Rerun the simulation for this new analysis model as before by going to the tab *Run*.

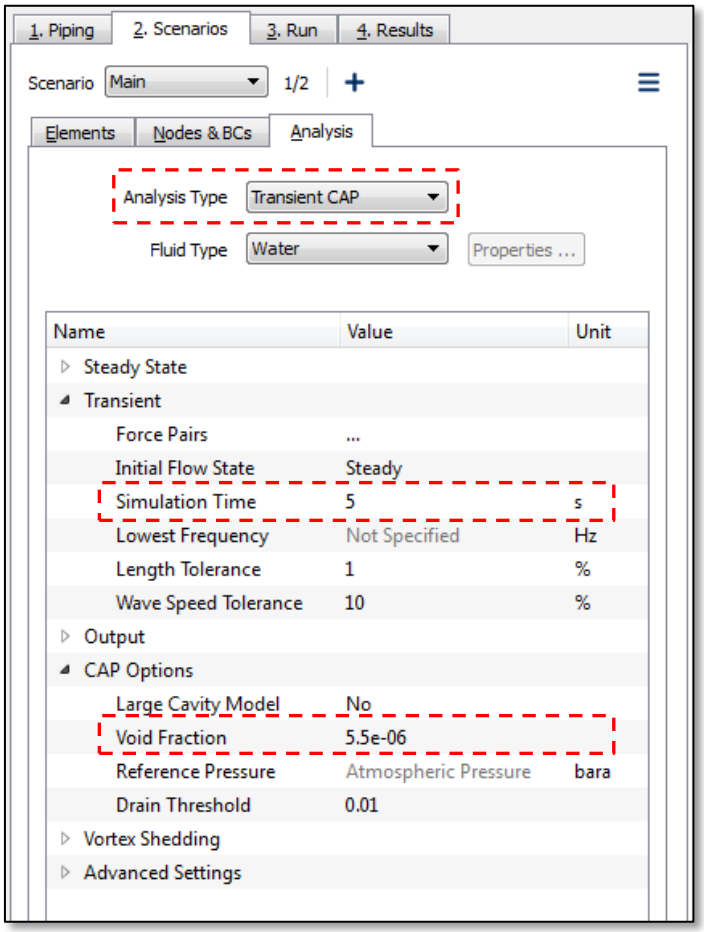

Figure 15 | Input screen for a transient CAP analysis

Copyright © Dynaflow Research Group.

4.3.2. Results for the transient CAP analysis

In the tab *Results* the results for the transient pressure variation can be studied again in the 2-D Output, see Figure 16.

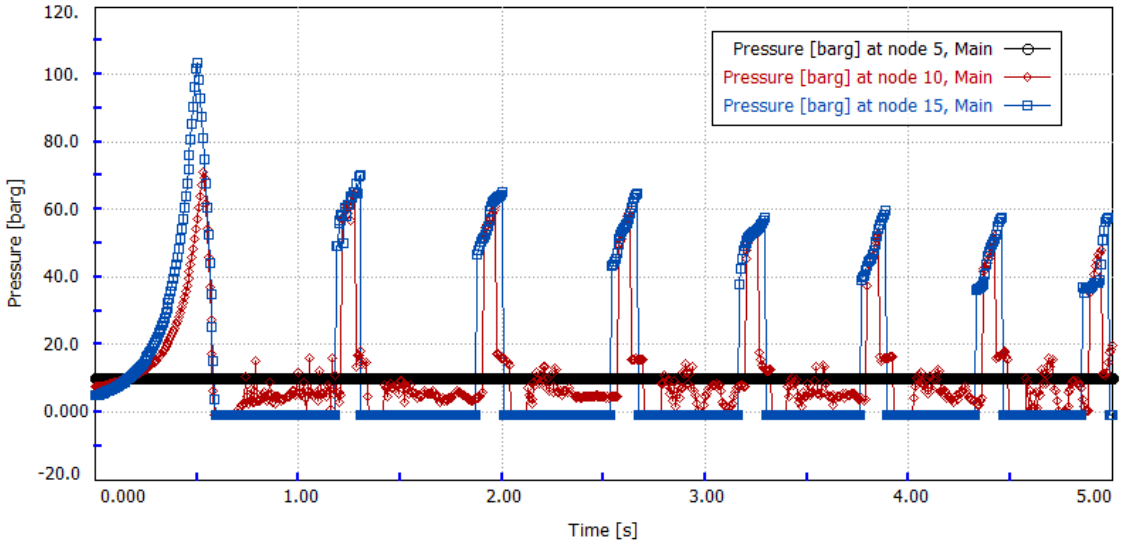

Figure 16 | Results for the *Pressure* variation through time for the transient CAP analysis

Reviewing Figure 16, the results look better this time. At the moment where the pressure in Figure 14 dropped below the vapour pressure (after 0.6 seconds), the pressure now remains constant at -0.99 barg (0.01 bar above vacuum). Also the longer simulation shows more of the transient events taking place after the valve is completely closed.

The user is able to interact with the plotted results. By clicking on or in the vicinity of a data point, the value of the selected data point is displayed. Similarly, using a click and drag movement, the user can create a view window to zoom into the selected region (return to initial view by double clicking inside the plot area). Likewise, the legend can be moved to a different location, by clicking and holding the legend and dragging it across the plot.

To change the settings of the plotted results, select *View* → *Settings*, or click the settings button ⚙. Several options are available to adjust the display of the plot including; title, range of the axes, labels, and plot style.

To export the plot or results, select *File* → *Save As*, or click the button 💾. The user is able to select a variety of different file type, to either export the figure or the data set. Select the file extension *PNG Image*, and save the file in the current working directory with the file name *PressurePlot*.

To see the effect of a higher inlet pressure on the transient flow rate, select *Node* and enter *10* in the 2-D Output and plot the *Flow Rate* for both scenarios. Go to *Settings* and change the range of the x-axis to 1 second. The results should look like Figure 17. It can be seen that for

Copyright © Dynaflow Research Group.

both scenarios the flow rate becomes negative (back flow) shortly after the valve completely closes. After 0.7 seconds the wave front reflected from the BC at node 5 passes node 10 to increase the flow rate again.

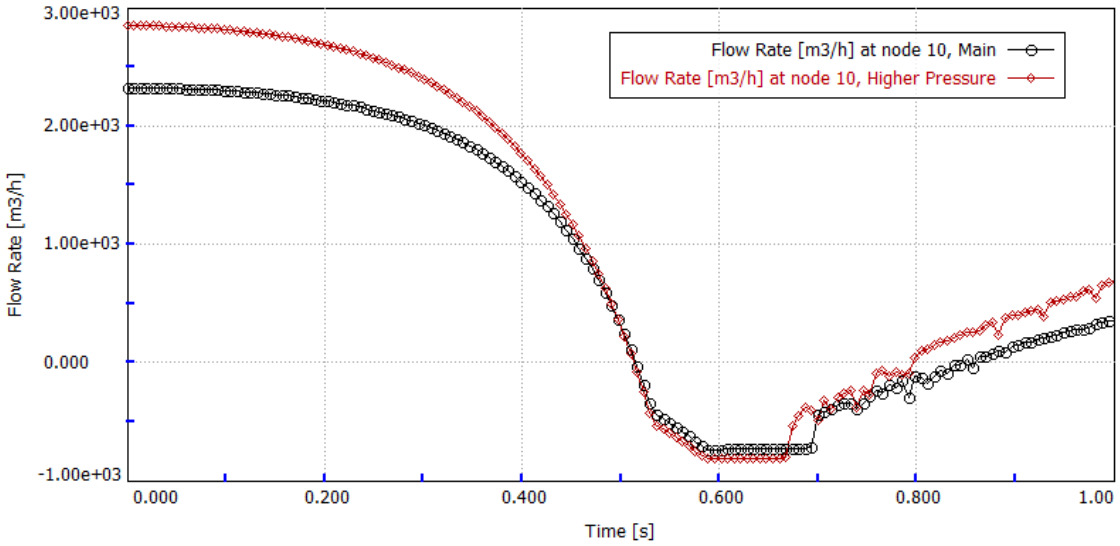

Figure 17 | Results for the *Flow Rate* variation at node 10 through time for both scenarios

Results for the flow rate through the valve can be reviewed at *2-D output* by selecting *Element, V_Shutoff* and plotting the *Flow Rate* and *Opening* for the *Main* scenario. As expected, the flow rate decreases in time while closing the valve, see Figure 18.

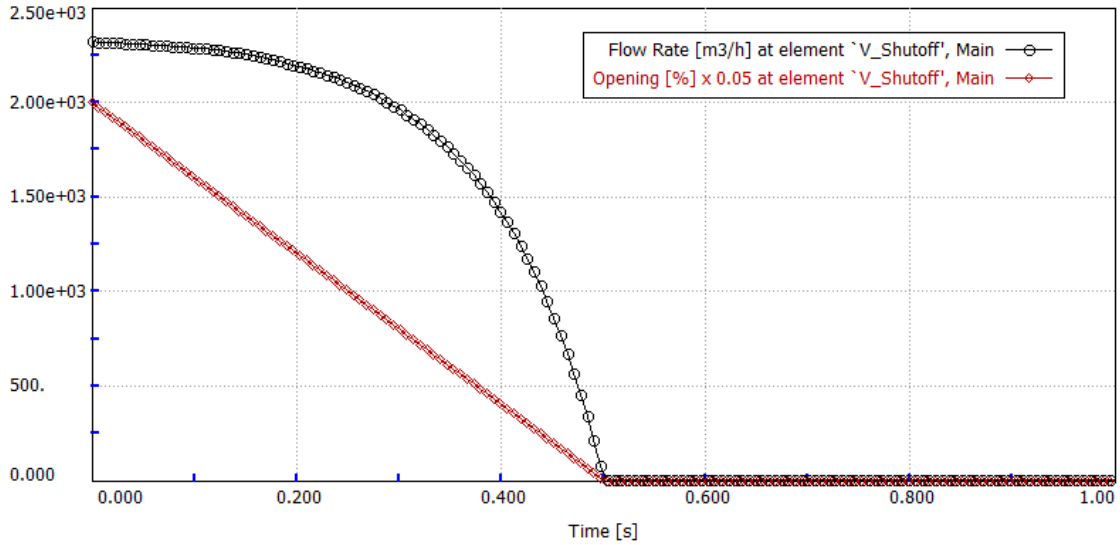

Figure 18 | Results for the *Flow Rate* through the valve and the *Opening* while closing

Copyright © Dynaflow Research Group.

5. THEORETICAL BACKGROUND

The transient behaviour of a system as discussed in this example, depends on the specific acoustic response time of the system and the duration of the transient event. When the duration of the transient event is much larger than the response time, the system will respond in a "slow" or pseudo-static mode. When the transient event duration is less than the response time, the system will respond in a "fast" or dynamic mode.

The system acoustic response time can be found from the approximate expression:

$$2L/c$$

Where L is the distance from the source of the disturbance to the point in the piping system where the disturbance will be reflected, and "c" is the speed of sound in the fluid. For rough estimates of the response time in liquid filled pipes c may be taken as 1000 meters/second (approximately 3000 feet per second), and the basic *system acoustic response time* can be found from:

$$L / 500 \text{ [sec.]} \qquad \text{with L in meters}$$

or:

$$L / 1500 \text{ [sec.]} \qquad \text{with L in feet}$$

The system acoustic response time for the example can be found from:

$$\text{time} = (2 \times 80 \text{ m}) / (1000 \text{ m/s}) = 0.16 \text{ seconds.}$$

The transient event duration in the example is 0.5 seconds, which is larger than the system response time, so "slow" responses will be observed. The increasing pressure wave will return to the source and begin cancelling further increases before the full value of the surge pressure will be applied to the system.

This phenomenon can also be experienced in a household when a water tab is shut quickly. Then a resounding "water hammer" is heard in the pipes. But when the water tab is slowly closed, there is hardly a whisper.

The maximum pressure rise that could occur in any system due to a theoretical "instantaneous" valve closure is:

$$\text{pressure rise} = (\text{fluid density})(c)(V)$$

Where (c) is the speed of sound in the fluid, and (V) is the initial velocity of the fluid.

BOSfluids calculates the initial velocity of the fluid in the steady state solution. For the example, analyzed above the initial velocity is 14 meters/sec.

Copyright © Dynaflow Research Group.

Instantly stopping this flow would produce pressure rises in the order of:

$$\text{pressure rise} = (1000\ \text{kg/m}^3)(1000\ \text{m/s})(14\ \text{m/s}) = 14{,}000{,}000\ \text{N/m}^2$$

This is equivalent to 140bar. The BOSfluids calculation showed a maximum pressure rise in the system of 100bar, which is less than the theoretical pressure rise for an instantaneous valve closure, but still considered large. This is due to the relative fast closure of the valve (even though the system was considered "slow" in terms of response time) and the large velocity. To considerably reduce the pressure rise for this system a larger valve closure time should be applied. A more realistic value for this typical example would be a valve closure time of 10 seconds.

Copyright © Dynaflow Research Group.